Airflow in a World Exposition Pavilion Studied by Scale-Model Experiments and Computational Flud Dynamics

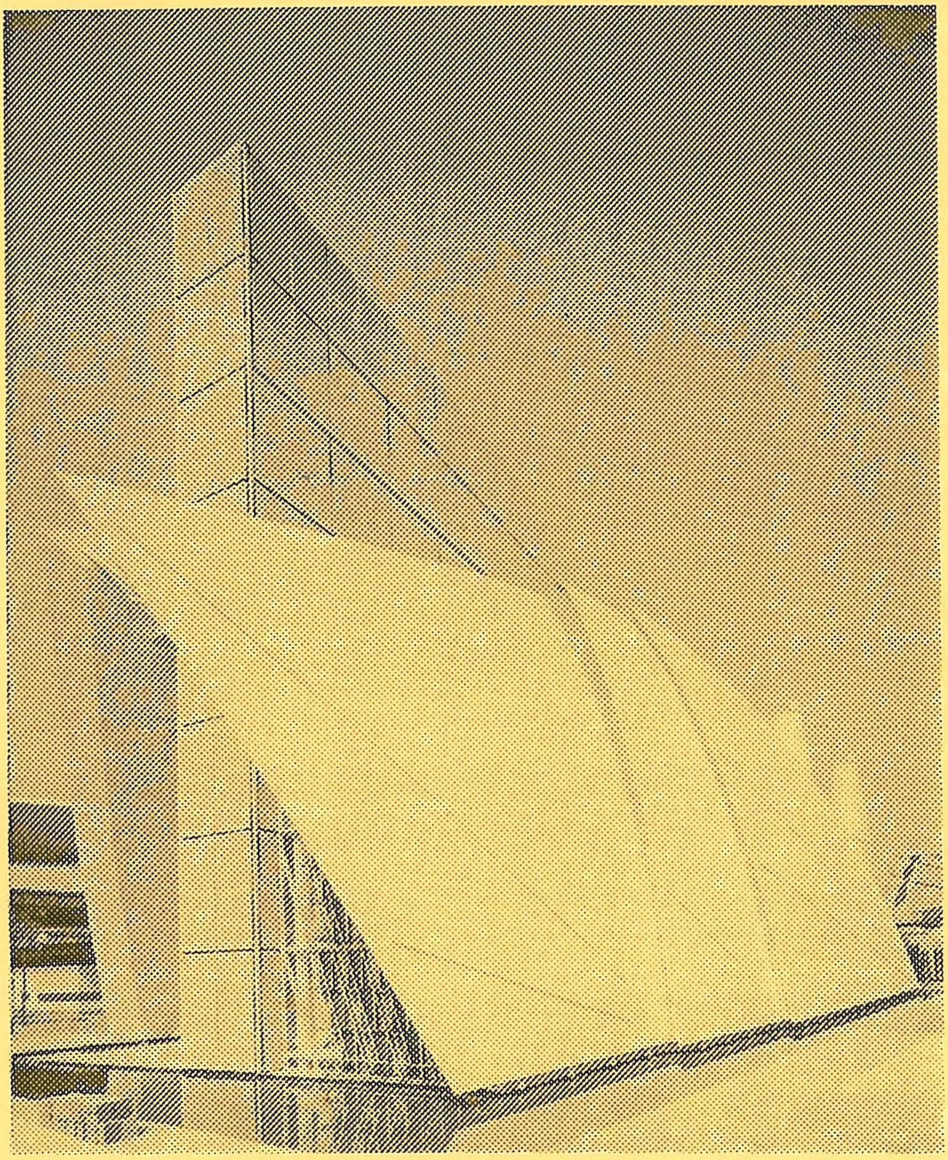
Nielsen, Peter V.

Publication date:
1998

Document Version
Publisher's PDF, also known as Version of record

INSTITUTTET FOR BYGNINGSTEKNIK

DEPT. OF BUILDING TECHNOLOGY AND STRUCTURAL ENGINEERING
AALBORG UNIVERSITET • AAU • AALBORG • DANMARK



INDOOR ENVIRONMENTAL ENGINEERING
PAPER NO. 84

Reprint from ASHRAE Transactions, Vol. 101, Part 2, 1995, pp. 1118-1126

P. V. NIELSEN

AIRFLOW IN A WORLD EXPOSITION PAVILION STUDIED BY SCALE-MODEL EXPERIMENTS AND COMPUTATIONAL FLUID DYNAMICS

AUGUST 1998 ISSN 1395-7953 R9825

The papers on INDOOR ENVIRONMENTAL ENGINEERING are issued for early dissemination of research results from the Indoor Environmental Engineering Group at the University of Aalborg. These papers are generally submitted to scientific meetings, conferences or journals and should therefore not be widely distributed. Whenever possible reference should be given to the final publications (proceedings, journals, etc.) and not to the paper in this series.

Printed at Aalborg University

INSTITUTTET FOR BYGNINGSTEKNIK

DEPT. OF BUILDING TECHNOLOGY AND STRUCTURAL ENGINEERING

AALBORG UNIVERSITET • AAU • AALBORG • DANMARK

INDOOR ENVIRONMENTAL ENGINEERING

PAPER NO. 84

Reprint from ASHRAE Transactions, Vol. 101, Part 2, 1995, pp. 1118-1126

P. V. NIELSEN

AIRFLOW IN A WORLD EXPOSITION PAVILION STUDIED BY SCALE-MODEL EXPERIMENTS AND COMPUTATIONAL FLUID DYNAMICS

AUGUST 1998 ISSN 1395-7953 R9825

SD-95-14-2

AIRFLOW IN A WORLD EXPOSITION PAVILION STUDIED BY SCALE-MODEL EXPERIMENTS AND COMPUTATIONAL FLUID DYNAMICS

Peter V. Nielsen, Ph.D.
Member ASHRAE

ABSTRACT

The ventilation design concept, model experiment results, two-dimensional computational fluid dynamics simulation, and on-site measurements are presented for the Danish Pavilion project at the 1992 World Exhibition in Seville.

The paper gives a short project history for the building and the ventilation system. The air-conditioned restaurant and exhibition hall have a floor area of 450 m^2 (4,850 ft^2) and a mean height of 18 m (59 ft). It is ventilated by a "cooling wall" with a height of 12 m (39 ft). The flow from the cooling wall is similar to the flow in a room with displacement ventilation. Scale-model experiments and computational fluid dynamics simulations indicate a velocity level in the occupied zone of approximately 0.6 m/s (~ 120 fpm) and the results are confirmed by on-site measurement in the exhibition hall. It is shown that even a simplified two-dimensional flow simulation will give valuable information to be used in the design procedure.

INTRODUCTION

Large ventilated air spaces, such as shopping arcades, atria, and exhibition buildings, have become popular in the last decade. The main purpose of designing the air distribution in such constructions is to obtain control of the energy flow and the temperature level. It is very important to have a high ventilation efficiency in the occupied areas and a system that can handle this area without a large air exchange in the rest of the air volume. Smoke movement in case of a fire and necessary escape routes are other important subjects. It is also necessary to limit the air velocity in the occupied areas because people may work at a restricted activity level in the shops and in open offices.

It is not possible to use full-scale experiments in the design of the air distribution system due to large dimensions. It may also be difficult to use simplified design methods such as those based on throws of jets and penetration depths of nonisothermal jets. This is due to the complicated geometry that is present in some situations. Several sources of air movement, such as diffusers, pressure difference around the building, cold downdraft, and thermal plumes, also make it difficult to use simplified methods.

Scale-model experiments (SME) and computational fluid dynamics (CFD) are two possible methods for the determination of the air distribution system. They are discussed in connection with the design of an air distribution system for the exhibition hall in the Danish World Exposition Pavilion. The results are further compared with on-site measurements of air velocities obtained in the building by full-scale tests.

The Danish pavilion concept was developed in 1989 and entered the design phase in January 1990. In August 1991 the complete prefabricated building was shipped from Denmark to Spain. The erection in Seville was complete in November 1991 and in March 1992 all services were functioning. On April 20, 1992, the World Exposition "EXPO '92" opened and ran until October 12, 1992. The Danish Pavilion was later bought by Tamba State in Japan and dismantling of the pavilion was completed in December 1992. The pavilion arrived in Japan in February 1993. A further description is given in DAP (1992).

THE BUILDING AND THE VENTILATION SYSTEM

The building has two main elements: a steel-framed "container" structure, facing west, with a floor area of 45.0 m by 2.5 m (147 ft by 8.2 ft), a height of 24 m (79 ft), and a fiberglass "sail" construction, facing east, which leans against the container structure (see Figure 1). The exhibition room is thus formed between the "sails" and the "containers" and it is enclosed by glass walls to the north as well as to the south. It has an average height of 18 m (59 ft), a length of 45 m (147 ft), an average width of 10 m (33 ft) at floor level and, furthermore, it has a triangular cross section.

It was very quickly determined that a traditional air-conditioning system for this exhibition room could not be accepted neither architecturally nor economically. Numerous alternatives and ideas were investigated, and the nature of the pavilion and the concept of a world exposition encour-

Peter V. Nielsen is a professor in the Department of Building Technology and Structural Engineering at Aalborg University, Aalborg, Denmark.

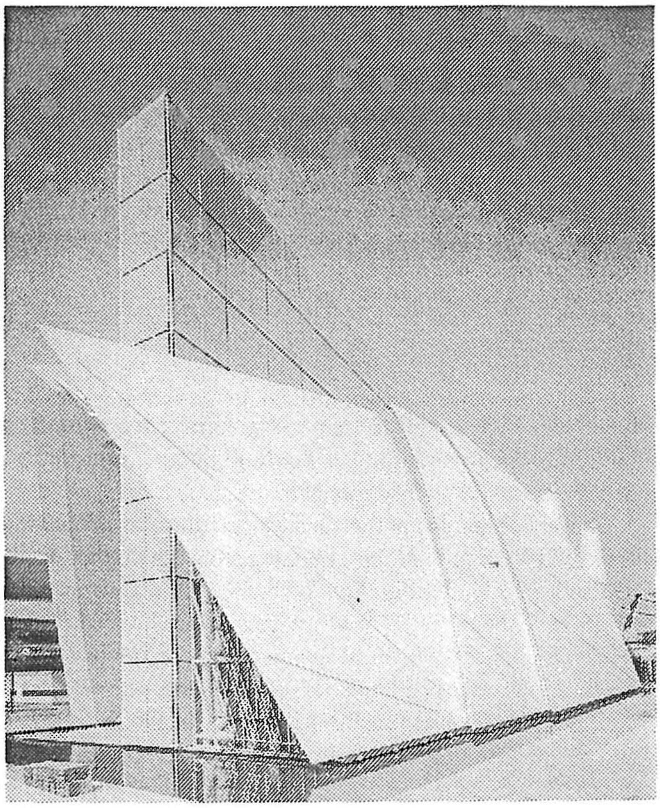


Figure 1 *The Danish Pavilion at the World Exposition EXPO '92 in Seville.*

aged the pursuit of experiments. The selected design was based simply on installation of exposed cooling elements in the south gable of the room and an extract fan in the top north end of the room (see Figure 2). Air is drawn through the cooling elements, where it cools down and flows through the occupied zone (Fox and Nielsen 1993).

The design load of the occupied zone in the restaurant and exhibition hall is 48 kW (164,000 Btu/h), corresponding to 300 persons in the pavilion. Slide and video equipment is installed above the occupied zone. This equipment will generate another 130 kW (444,000 Btu/h), which is expected to move upward in convective flows, thus causing a high temperature in the upper part of the pavilion. This part of the heat load will be extracted directly by the upper ventilator at a high temperature effectiveness.

The heat load transmission from outside will be small because one side of the exhibition hall is an air-conditioned building and the other side is a fiberglass sail with external water cooling. The north wall is a glass window and the south wall—where the solar radiation is present—is the supply wall with cooling devices.

The cooling devices in the south gable are designed to give an inlet air temperature of 26°C (79°F) at an outdoor temperature of 38°C (100°F). The exhaust fan in the ceiling, which is a part of the fire ventilation, is designed for a flow rate of 10 m^3/s (21,000 cfm), corresponding to a temperature difference between return and supply of 4 K (7°F) due to the load in the occupied zone. The load from slide and video equipment will increase this temperature difference.

It is important to restrict the air velocities in the occupied zone. A velocity up to 1 m/s (200 fpm) is acceptable instead of the normal limiting figure of 0.15 m/s (30 fpm) due to the high air temperature in the hall. The conditions are in good agreement with the thermal comfort conditions of 0.3 clo and 1 met (Fanger 1967).

SCALE-MODEL EXPERIMENTS

The following section discusses the theory behind scale-model experiments and shows the results obtained from the experiments.

Dimensionless Numbers and Similarity Principle

It can be shown from theory that a similar flow will be obtained in a full-size room and in a scale model if the dimensionless boundary conditions are identical and if the Archimedes number, the Reynolds number, and the Prandtl number are the same in both situations.

It is impossible to make a scale-model experiment in a strongly reduced scale if all the dimensionless numbers have to be kept constant. If, for instance, the scale is reduced by a factor of 10, then the velocity has to be increased by a factor of 10 due to the Reynolds number, which will give an increase in the temperature difference by a factor of 1,000 in order to keep the Archimedes number. The Prandtl number

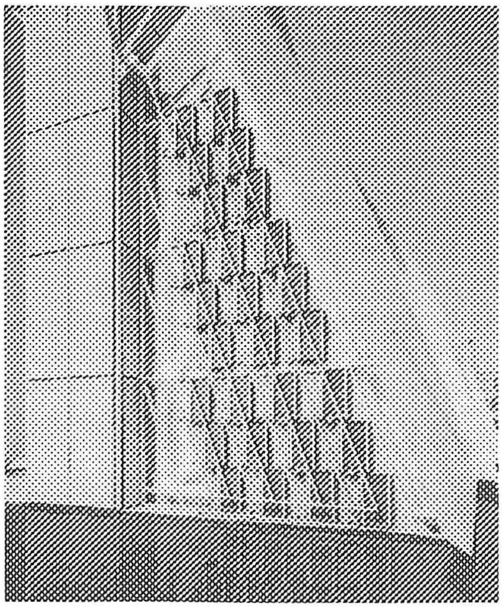


Figure 2 *Cooling elements in the south gable of the pavilion.*

is, on the other hand, unchanged when air is used as the fluid in the model experiments.

The problem can, in practice, be overcome if the Reynolds number is high and the flow pattern is governed mainly by fully developed turbulence. It is possible to ignore the Reynolds number and the Prandtl number at a sufficiently high level of velocity because the structure of the turbulence and the flow pattern will be similar at different supply velocities and therefore independent of the Reynolds number. The transport of thermal energy by turbulent eddies will also dominate the molecular diffusion and will therefore be independent of the Prandtl number. (See Nielsen [1993] for a further discussion of scale-model experiments.)

Measurements

The model experiments for the restaurant and exhibition hall were conducted in a model with the scale 1/10. The model room was made with the correct geometrical shapes with two nonparallel sidewalls, which gave a triangular vertical cross section perpendicular to the main flow direction and an increased width of the occupied zone in the downstream direction of the flow.

The supply wall with the cooling elements was simulated by a diffuser that covered the whole supply area. The velocities were measured by spherical omnidirectional probes specially developed for low-velocity applications. The probes had a measuring accuracy of ±0.02 m/s (±4 fpm) and only mean velocities were measured.

Sixteen different experiments were carried out and three of the more essential ones will be discussed in detail in the following section. Table 1 shows the main parameters.

In the experiment, the Archimedes number is based on the height, h, of the inlet device as the length scale and the face velocity, u_f, as the reference velocity where the face velocity is the supply flow divided by a reference area. The experiments only consider the heat load in the occupied zone.

Figure 3 shows the normalized velocity distribution in the room as a function of the Archimedes number. It is quite obvious that the flow is a stratified flow with the highest velocity in the occupied zone. Smoke measurements show that the cold air from the cooling device moves horizontally along the floor in the restaurant and exhibition section.

Measurements show that the velocity has a fairly constant level in the occupied zone, even far downstream from the wall with the cooling device, as shown in Figure 3. The flow is plane and it is a general experience that the velocity in a plane stratified flow is constant and independent of distance from the inlet device (Nielsen 1994a).

Figure 4 shows the maximum velocity in the stratified flow. The velocity increases for increasing Archimedes number corresponding to a decreasing thickness of the flow. The level of the maximum velocity, u_m/u_f, in the occupied zone is 1.1 in test A, 1.5 in test B, and 1.8 in test C. The measured velocity close to the supply opening may be slightly influenced by the actual diffuser used in the scale model.

The influence from the location of the return opening was tested by using the lower opening alone, the upper opening alone, and a combination of both openings with 50% flow in each opening. It was not possible to measure any effect on the velocity level in the model. Experiments with a variable temperature at the surface of the supply device, with the highest temperature in the upper part of the device, did not show any pronounced influence on the velocity level.

Transformation of Model Experiments into Full Scale

The volume flow rate from the cooling device in the south wall is designed to be 10 m^3/s (21,000 cfm). This is equivalent to a face velocity, u_f, of 0.35 m/s (69 fpm). The Archimedes number in test A corresponds to a temperature difference between the return and supply devices of 1 K (1.8°F), corresponding to a heat load of 12 kW (41,000 Btu/h) in the restaurant and exhibition hall. This is a rather restricted heat load, and the model experiments show that the maximum velocity in the occupied zone will be 0.38 m/s (75 fpm) in full scale. A heavier load of 36 kW (123,000 Btu/h)—corresponding to test C—will give up to 0.63 m/s

TABLE 1
Conditions and Results for Three Model Experiments

	h m (ft)	ΔT_o K (°F)	u_f m/s (fpm)	Ar
Test A	1.2 (3.9)	5.6 (10.1)	0.26 (51.1)	3.35
Test B	1.2 (3.9)	6.0 (10.8)	0.175 (34.4)	7.87
Test C	1.2 (3.9)	7.5 (13.5)	0.175 (34.4)	9.84

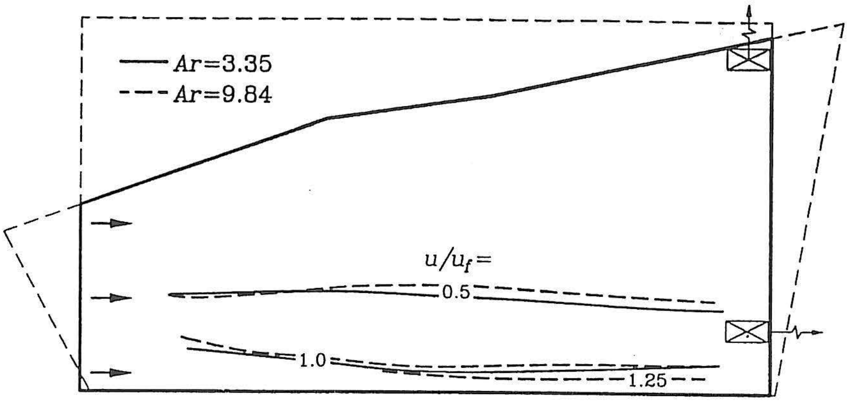


Figure 3 *Normalized velocity distribution vs. Archimedes number in the exhibition hall.*

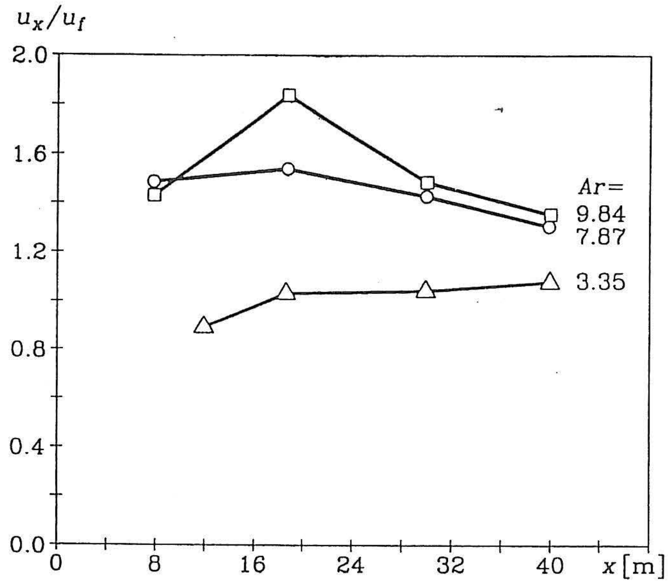


Figure 4 *Maximum velocity in the stratified flow vs. distance from the supply wall.*

(124 fpm) in the occupied zone. An increase of the flow rate to 14.8 m^3/s (31,400 cfm) will make it possible to handle a load of 40 kW (137,000 Btu/h) in the occupied zone with a maximum velocity of 0.57 m/s (112 fpm), as shown in Table 2.

COMPUTATIONAL FLUID DYNAMICS

The fluid dynamics research is strongly influenced by the increasing computer power of the last decades. Due to this, the cost for a given job will decrease by a factor of 10 during every eighth year. The development shows not only decreasing costs but decreasing computer time. There are more reasons for this development. First, computer efficiency is increasing more rapidly than computer costs and this tendency seems to continue. Second, the flexibility of different software as pre- and post-processor software occurs and, furthermore, new software continues to develop. Improvements in the fundamental routines (e.g., the grid generation procedure and the numerical method) also contribute to increasing speed.

These tendencies have also influenced indoor environmental technology. Today, a number of codes developed at universities and commercially developed codes are available for the prediction of air distribution in rooms. This section will show the predictions obtained by a simple two-dimensional code to give an example of the information that can be obtained by using such a method compared with scale-model experiments.

A general review of CFD applications within indoor environmental technology is given in Christianson (1989), and current status and capabilities are addressed by Jones and Whittle (1992). The prospects for CFD in room air contaminant control have recently been reviewed by Nielsen (1994b).

TABLE 2
Transformation of Scale-Model Experiments into Full-Scale Conditions

	q_o m^3/s (cfm)	ΔT_o K (°F)	u_f m/s (fpm)	u_m m/s (fpm)	Q W/m^2 (Btu/hft^2)
Test A	10.0 (21000)	1.0 (1.80)	0.35 (68.9)	0.38 (75)	27.4 (8.70)
Test C	10.0 (21000)	3.0 (5.40)	0.35 (68.9)	0.63 (124)	82.2 (26.10)
Test A	14.8 (31400)	2.2 (3.96)	0.52 (102.4)	0.57 (112)	91.3 (29.00)

GOVERNING EQUATIONS AND NUMERICAL METHOD

The air movement in a room with recirculation flow and temperature differences is described by the transport equations for mass, momentum, and energy. It is widely used to describe the turbulence by the eddy viscosity concept. The eddy viscosity, μ_t, can be predicted from a k-ε turbulence model consisting of two transport equations for turbulent kinetic energy, k, and dissipation of turbulent kinetic energy, ε (Launder et al. 1973).

The k-ε model is only valid for a fully developed turbulent flow. This is supposed to be the case in this situation with very large dimensions, as discussed in the last section on scale-model experiments. The CFD method can be used to make a review of the level of the turbulence. This will be discussed in the appendix.

Low turbulence effects can be predicted in near-wall regions with a low Reynolds number model (LRN), as given by Launder and Sharma (1978). However, the model is not suitable in a general form for prediction of turbulence far from the surfaces.

The transport equations are reformulated into finite-difference equations and solved by an iteration procedure. A detailed description of the method and especially of the finite-volume concept is given by Patankar (1980). Baker et al. (1993) show the application of a finite-element method for the prediction of room air distribution.

Boundary Conditions

It was decided to make the CFD predictions in a two-dimensional Cartesian code to study the level of information that can be obtained in this simplified situation in comparison with scale-model experiments. The code is based on time-averaged transport equations for momentum, mass, and energy and a k-ε turbulence model. The numerical method uses the finite-volume concept and a staggered grid.

Figure 5 shows the boundary conditions for the geometry. The room has the height of 18 m (59 ft), and 12 m (39 ft) of the south wall are covered by the supply device. The return opening is located in the north corner of the ceiling. The real exhibition hall has a triangular shape perpendicular to the direction of the main flow. This is impossible to take into consideration in this case with a two-dimensional configuration of the CFD method. It is also obvious from Figure 5 that the geometry of the ceiling regions is simplified in the CFD predictions.

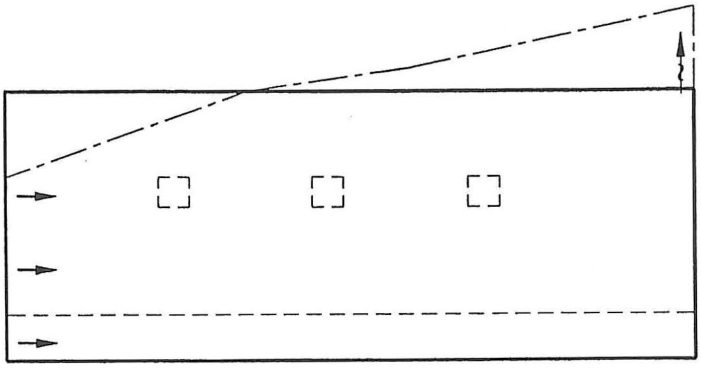


Figure 5 *Boundary conditions for two-dimensional CFD predictions. The dotted lines show the locations of volume heat sources.*

The locations of the heat sources are shown in Figure 5. The heat load from the occupied zone is described as a volume source with a height of 3 m (10 ft). One prediction (case D) also takes account of additional heat sources with a higher location, as shown by means of the three volumes in the figure.

The two-dimensional predictions are made for full-scale situations corresponding to test A and test C in order to facilitate comparisons with the scale-model tests. Table 3 shows that the face velocity, u_f, is used for the description of the mass flow rate, which means that the flow rate per supply area is identical in predictions and in full scale. The heat load per floor area is also identical in predictions and in full scale. The assumptions have the consequences that the

TABLE 3
Boundary Conditions for the CFD-Predictions

	h m (ft)	u_f m/s (fpm)	Q W/m^2 (Btu/hft^2)	Ar Full scale	Ar_{2D} 2D Prediction
Case A	12 (39)	0.35 (68.9)	27.4 (8.70)	3.35	0,78
Case C	12 (39)	0.35 (68.9)	82.2 (26.10)	9.84	2.35
Case D	12 (39)	0.35 (68.9)	27.4 + 296.7 (8.70 + 94.20)		9.29

Archimedes numbers in full-scale flow are different from the Archimedes numbers in the two-dimensional predictions because the effective supply areas are slightly different and, especially, because the real hall has a triangular shape that corresponds to a low "velocity" in the Archimedes number. It is important to work with the correct flow rates of energy and momentum and therefore it is advisable to use the boundary conditions in Table 3 when the results are compared with the measured flow fields in test A and test C.

Predictions

The predicted flow field is unidirectional in the main part of the hall, including the occupied zone, and it has a small recirculation above the supply opening. The maximum velocity is located at the height of 0.5 to 1.0 m (1.6 to 3.3 ft), which is similar to the relative location measured in the model experiments.

Figure 6 shows the maximum velocity in the stratified flow versus distance from the south wall with the supply opening. The velocity level for the two Archimedes numbers is almost identical to the velocities measured in the scale model (see Figure 4). The scale-model experiments obtain the highest velocities close to the supply opening, while the CFD predictions have the highest velocities in the opposite part of the hall. This may partly be explained by the fact that the real hall is narrow close to the supply wall (~ 6.2 m [~ 20 ft]) and wide in the other end at the north wall (~ 13.4 m [~ 44 ft]) and partly by the fact that the velocities measured in the scale model close to the supply opening may be influenced by local velocity gradients generated by the actual diffuser in the model.

The maximum velocity in the occupied zone is 0.41 m/s (81 fpm) in case A and 0.56 m/s (110 fpm) in case C. The full-scale levels of the maximum velocity in scale-model tests A and test C are 0.38 m/s (75 fpm) and 0.63 m/s (124 fpm), respectively. Comparisons between SME and CFD show an accuracy that is far above an expected level when all the simplifications in the boundary conditions are taken into consideration.

It is generally expected that the flow in the occupied zone is unaffected by highly placed heat sources. The situation is examined in case D, where the slide and video equipment is simulated as three sources with a total heat flux of 130 kW (444,000 Btu/h) situated as indicated in Figure 5. Comparisons with case A show that the maximum velocity in the occupied zone has the same level (0.43 m/s [85 fpm]) but the location is now in the south end of the hall.

Figure 7 shows the temperature profiles at x = 16 m (52 ft), which is between two of the heat sources. Both profiles show that the temperature in the occupied zone is uninfluenced by the highly placed volume sources. The additional heat sources add a stratification to the flow but this does not influence the temperature in the occupied zone. The predictions are made without radiation from the ceiling. There must be some effect in practice but it is restricted by the external water cooling of the building surface.

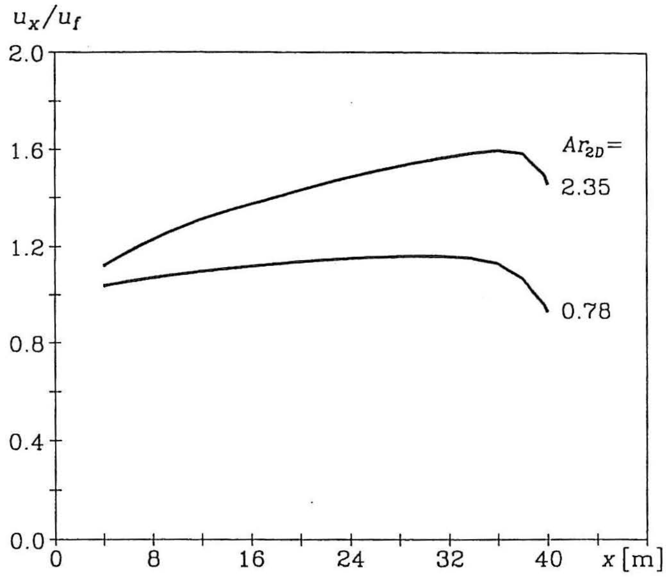


Figure 6 *Maximum velocity in the flow vs. distance from the supply wall.*

The appendix shows that the flow can be expressed as a fully developed turbulent flow. The turbulence is mainly generated at the supply wall and not by velocity gradients in the room. It is therefore the design of the supply devices that dictates the level of the turbulence, and it is the dimensions and velocity at the supply wall that give the characteristic Reynolds number.

All the predictions given in this section are obtained from a typical commercial code. The specification and computing time are low, partly due to an efficient front-end code

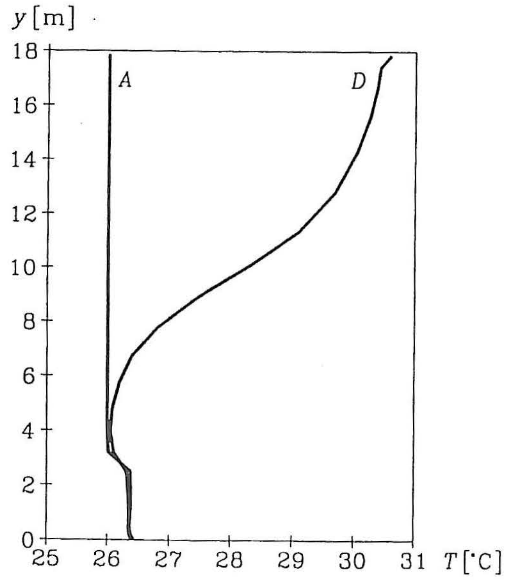


Figure 7 *Vertical temperature profiles at x = 16 m (52 ft) for case A and case D.*

and partly due to the use of a two-dimensional flow simulation. The boundary conditions correspond to an unstable situation (low location of a heat source and cold flow above) and this affects the stability of the convergence. It is necessary to have some experience in CFD in connection with the use of underrelaxation on the finite-difference equations.

FULL-SCALE ON-SITE MEASUREMENTS

On-site measurements show air velocities at a level of 0.8 m/s (157 fpm), roughly corresponding to the velocity level obtained by the model experiments (see Table 4). The velocity has a quicker reduction than expected downstream in the occupied zone, probably due to the influence from the highly placed heat sources. (This effect was also observed in the case D simulations.) The measurements could only take place in the early morning of May 6, 1992, when the outdoor temperature was 19.7°C (67.5°F) and the inlet air temperature was 12.1°C (53.8°F). Show projectors and lighting were at full power, whereas occupant loading was light (approximately 20 people). Discharge air temperature was 27.5°C (81.5°F), indicating the expected strong vertical stratification corresponding to a high ventilation effectiveness. The Archimedes number was approximately 50 (Fox and Nielsen 1993).

The conclusions from on-site measurements are (a) uniform temperature distribution in the occupied zone with a small horizontal temperature gradient, (b) strong vertical stratification ensuring separation between occupied zone loads and other room loads, and (c) air velocities equal to those predicted by scale-model experiments and CFD.

TABLE 4
Mean Temperature and Maximum Velocity Measured Across the Room vs. Distance from the Supply Wall (On-site measurements 1 m [3.3 ft] above floor level)

Distance	Temperature, °C (°F)
2 m (6.5 ft)	12.7 (54.9)
16 m (52.5 ft)	14.1 (57.4)
24 m (78.7 ft)	14.6 (58.3)
32 m (105.0 ft)	14.9 (58.8)
40 m (131.2 ft)	14.7 (58.5)

Distance	Velocity, m/s (*ft*)
2 m (6.5 ft)	0.6 (118)
3 m (9.8 ft)	0.6 (118)
5 m (16.4 ft)	0.8 (157)
8 m (26.2 ft)	0.5 (98)
12 m (39.4 ft)	0.1 (20)

CONCLUSIONS

Scale-model experiments (SME) and computational fluid dynamics (CFD) simulations are important design methods.

The air movement in large enclosures can be investigated in a scale model. Special attention must be paid to the low-turbulence effect in SME. Large dimensions and large Reynolds numbers will ensure high-turbulence flow, and the Archimedes number is the only important parameter in the experiments.

SME shows that the flow in the exhibition hall is unidirectional and stratified. The experiments indicate a velocity level of approximately 0.6 m/s (120 fpm) through the whole occupied zone.

CFD prediction of air movement in large enclosures is a very important alternative to scale-model experiments. The turbulence model in a CFD method is based on fully turbulent flow (large Reynolds number) and this is easily obtained in a large enclosure.

CFD predictions confirm the general flow pattern obtained by SME. A velocity level of approximately 0.6 m/s (120 fpm) is indicated by two-dimensional flow simulation. It is possible to show that the flow in the occupied zone is unaffected by heat sources in the upper part of the hall.

Specification and computing times are very low for two-dimensional flow simulation. Therefore, it may be very efficient to include two-dimensional flow simulation at an early stage in the design phase because the predictions are valuable for further design work. Skill and experience are required for the specification work and handling of convergence problems.

On-site measurements in full scale show that the temperature distribution in the occupied zone only has a small horizontal gradient. There is a strong vertical stratification, which ensures separation between the occupied zone load and other room loads. The air velocity level in full-size measurements is equal to the level obtained by SME and CFD.

NOMENCLATURE

Ar	=	Archimedes number
Ar_{2D}	=	Archimedes number for two-dimensional predictions
c_μ	=	constant in turbulence model
f_μ	=	variable in turbulence model
f_2	=	variable in turbulence model
h	=	height of supply wall
k	=	turbulent kinetic energy
q_o	=	flow rate
Q	=	heat load per floor area
R_t	=	local turbulent Reynolds number
u_f	=	face velocity
u_m	=	maximum velocity in the occupied zone
u_o	=	supply velocity
x	=	horizontal distance from supply wall
ΔT_o	=	temperature difference
ε	=	dissipation of turbulent kinetic energy
μ	=	molecular viscosity
μ_t	=	turbulent viscosity

REFERENCES

Baker, A.J., P.T. Williams, and R.M. Kelso. 1993. Development and validation of a robust CFD procedure for predicting indoor room air motion. *Proc. of the 6th International Conference on Indoor Air Quality and Climate, INDOOR AIR '93*, vol. 5, Helsinki, pp. 183-188.

Christianson, L.L., ed. 1989. *Building systems: Room air and air contaminant distribution.* Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.

DAP. 1992. *Danish pavilion—EXPO '92, Seville.* Aalborg: The Danish Architectural Press.

Fanger, P.O. 1967. Calculation of thermal comfort: Introduction of a basic comfort equation. *ASHRAE Transactions* 73(2).

Fox, S.G., and P.V. Nielsen. 1993. Model experiments in 1990 and on-site validation in 1992 of the air movement in the Danish Pavilion in Seville. *Proc. of the 6th International Conference on Indoor Air Quality and Climate, INDOOR AIR '93*, vol. 2, Helsinki, pp. 675-680.

Jones, P.J., and G.E. Whittle. 1992. Computational fluid dynamics for building air flow prediction—Current status and capabilities. *Building and Environment* 27(3): 321-338.

Launder, B.E., and B.I. Sharma. 1978. *Letters in heat and mass transfer* 1: 129.

Launder, B.E., D.B. Spalding, and J.H. Whitelaw. 1973. Turbulence models and their experimental verification. Imperial College, Heat Transfer Section, Reports HTS 73.

Nielsen, P.V. 1993. Model experiments for the determination of airflow in large spaces. *Proc. of the International Conference, INDOOR AIR '93*, vol. 5, Helsinki, pp. 253-258.

Nielsen, P.V. 1994a. Stratified flow in a room with displacement ventilation and wall-mounted air terminal devices. *ASHRAE Transactions* 100(1): 1163-1169.

Nielsen, P.V. 1994b. Prospects for computational fluid dynamics in room air contaminant control. *Proceedings of the 4th International Symposium on Ventilation for Contaminant Control, Ventilation '94*, part 1, pp. 26-46. Stockholm: Center for Research on Occupational Health.

Patankar, S.V. 1980. *Numerical heat transfer and fluid flow.* New York: Hemisphere Publishing.

APPENDIX

Launder and Sharma (1978) work with a low-Reynolds-number (LRN) model, which, among other things, contains the coefficients f_μ and f_2. The coefficients make a modification of the turbulence model in case of a low local turbulence Reynolds number R_t:

$$f_\mu = \exp(-3.4/(1 + R_t/50)^2) \tag{A1}$$

and

$$f_2 = 1 - 0.3\exp(-R_t^2)\,. \tag{A2}$$

A high turbulence k-ε model, which is used in this paper, may be considered a special version of an LRN model with f_μ and f_2 equal to 1.0. Those conditions are fulfilled in practice when R_t exceeds 400 everywhere in the flow domain, as seen from Equations A1 and A2.

The local turbulent Reynolds number can be expressed by

$$R_t = \frac{1}{c_\mu f_\mu}\frac{\mu_t}{\mu} \tag{A3}$$

where μ_t is turbulent viscosity and μ is the molecular viscosity of the air. $R_t > 400$ and $c_\mu = 0.09$ corresponds to

$$\mu_t/\mu > 40. \tag{A4}$$

Figure 8 shows the distribution of μ_t/μ at $x = 16$ m (52 ft). Equation A4 is fulfilled at this vertical section as well as in the rest of the main flow. This may indicate that the flow is fully turbulent, which also is a necessary assumption for the predictions because the k-ε model has been used as a turbulence model. The turbulence is mainly generated at the supply wall, where it has a level of $\mu_t/\mu \sim 50$. This is a low level, taking the large dimensions into consideration, and it is much higher in many practical situations. Velocity gradients are one of the important sources of turbulence, and the upper part of Figure 8 shows an increased level that can be obtained in areas with recirculation.

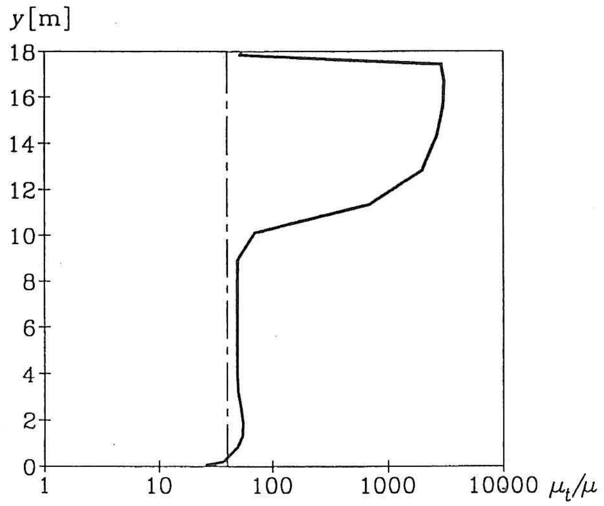


Figure 8 *Vertical turbulent viscosity profile at x = 16 m (52 ft) for case A.*

PAPERS ON INDOOR ENVIRONMENTAL ENGINEERING

PAPER NO. 51: Peter V. Nielsen: *Healthy Buildings and Air Distribution in Rooms.* ISSN 1395-7953 R9538.

PAPER NO. 52: Lars Davidson & Peter V. Nielsen: *Calculation of the Two-Dimensional Airflow in Facial Regions and Nasal Cavity using an Unstructured Finite Volume Solver.* ISSN 1395-7953 R9539.

PAPER NO. 53: Henrik Brohus & Peter V. Nielsen: *Personal Exposure to Contaminant Sources in a Uniform Velocity Field.* ISSN 1395-7953 R9540.

PAPER NO. 54: Erik Bjørn & Peter V. Nielsen: *Merging Thermal Plumes in the Indoor Environment.* ISSN 1395-7953 R9541.

PAPER NO. 55: K. Svidt, P. Heiselberg & O. J. Hendriksen: *Natural Ventilation in Atria - A Case Study.* ISSN 1395-7953 R9647.

PAPER NO. 56: K. Svidt & B. Bjerg: *Computer Prediction of Air Quality in Livestock Buildings.* ISSN 1395-7953 R9648.

PAPER NO. 57: J. R. Nielsen, P. V. Nielsen & K. Svidt: *Obstacles in the Occupied Zone of a Room with Mixing Ventilation.* ISSN 1395-7953 R9649.

PAPER NO. 58: C. Topp & P. Heiselberg: *Obstacles, an Energy-Efficient Method to Reduce Downdraught from Large Glazed Surfaces.* ISSN 1395-7953 R9650.

PAPER NO. 59: L. Davidson & P. V. Nielsen: *Large Eddy Simulations of the Flow in a Three-Dimensional Ventilated Room.* ISSN 1395-7953 R9651.

PAPER NO. 60: H. Brohus & P. V. Nielsen: *CFD Models of Persons Evaluated by Full-Scale Wind Channel Experiments.* ISSN 1395-7953 R9652.

PAPER NO. 61: H. Brohus, H. N. Knudsen, P. V. Nielsen, G. Clausen & P. O. Fanger: *Perceived Air Quality in a Displacement Ventilated Room.* ISSN 1395-7953 R9653.

PAPER NO. 62: P. Heiselberg, H. Overby & E. Bjørn: *Energy-Efficient Measures to Avoid Downdraft from Large Glazed Facades.* ISSN 1395-7953 R9654.

PAPER NO. 63: O. J. Hendriksen, C. E. Madsen, P. Heiselberg & K. Svidt: *Indoor Climate of Large Glazed Spaces.* ISSN 1395-7953 R9655.

PAPER NO. 64: P. Heiselberg: *Analysis and Prediction Techniques.* ISSN 1395-7953 R9656.

PAPER NO. 65: P. Heiselberg & P. V. Nielsen: *Flow Element Models.* ISSN 1395-7953 R9657.

PAPER NO. 66: Erik Bjørn & P. V. Nielsen: *Exposure due to Interacting Air Flows between Two Persons.* ISSN 1395-7953 R9658.

PAPER NO. 67: P. V. Nielsen: *Temperature Distribution in a Displacement Ventilated Room.* ISSN 1395-7953 R9659.

PAPER NO. 68: G. Zhang, J. C. Bennetsen, B. Bjerg & K. Svidt: *Analysis of Air Movement Measured in a Ventilated Enclosure.* ISSN 1395-7953 R9660.

PAPERS ON INDOOR ENVIRONMENTAL ENGINEERING

PAPER NO. 69: E. Bjørn, P. V. Nielsen: *Passive Smoking in a Displacement Ventilated Room.* ISSN 1395-7953 R9714.

PAPER NO. 70: E. Bjørn, M. Mattsson, M. Sandberg, P. V. Nielsen: *Displacement Ventilation - Effects of Movement and Exhalation.* ISSN 1395-7953 R9728.

PAPER NO. 71: M. Mattsson, E. Bjørn, M. Sandberg, P. V. Nielsen: *Simulating People Moving in Displacement Ventilated Rooms.* ISSN 1395-7953 R9729.

PAPER NO. 72: H. Brohus: *CFD-Simulation of Personal Exposure to Contaminant Sources in Ventilated Rooms.* ISSN 1395-7953 R9734.

PAPER NO. 73: H. Brohus: *Measurement of Personal Exposure using a Breathing Thermal Manikin.* ISSN 1395-7953 R9735.

PAPER NO. 74: H. Brohus, C. E. Hyldgaard: *The Use of Tracer Gas Measurements in Detection and Solution of Indoor Air Quality Problems in a Danish Twon Hall.* ISSN 1395-7953 R9736.

PAPER NO. 75: C. E. Hyldgaard, H. Brohus: *Detection and Solution of Indoor Air Quality Problems in a Danish Twon Hall.* ISSN 1395-7953 R9737.

PAPER NO. 76: C. Topp, P.V. Nielsen, P. Heiselberg: *Evaporation Controlled Emission in Ventilated Rooms.* ISSN 1395-7953 R9739.

PAPER NO. 77: P. Lengweiler, P.V. Nielsen, A. Moser, P. Heiselberg, H. Takai: *Deposition and Resuspension of Particles.* ISSN 1395-7953 R9740.

PAPER NO. 78: J. Richter Nielsen, P.V. Nielsen. K. Svidt: *Air Distribution in a Furnished Room Ventilated by Mixing Ventilation.* ISSN 1395-7953 R9742.

PAPER NO. 79: P.V. Nielsen: *Design of Local Ventilation by Full-Scale and Scale Modelling Techniques.* ISSN 1395-7953 R9743.

PAPER NO. 80: P. Heiselberg, K. Svidt, H. Kragh: *Application of CFD in Investigation of Ventilation Strategies for Improvement of Working Environment in a Waste Incineration Plant.* ISSN 1395-7953 R9745.

PAPER NO. 81: P. Heiselberg, C. Topp: *Removal of Airborne Contaminants from a Surface Tank by a Push-Pull System.* ISSN 1395-7953 R9746.

PAPER NO. 82: P. Heiselberg: *Simplified Method for Room Air Distribution Design.* ISSN 1395-7953 R9747.

PAPER NO. 83: L. Davidson, P. V. Nielsen: *A Study of Laminar Backward-Facing Step Flow.* ISSN 1395-7953 R9802.

PAPER NO. 84: P.V. Nielsen: *Airflow in a World Exposition Pavilion Studied by Scale-Model Experiments and Computational Fluid Dynamics.* ISSN 1395-7953 R9825.

PAPER NO. 85: P. V. Nielsen: *Stratified Flow in a Room with Displacement Ventilation and Wall-Mounted Air Terminal Devices.* ISSN 1395-7953 R9826.

Department of Building Technology and Structural Engineering
Aalborg University, Sohngaardsholmsvej 57. DK 9000 Aalborg
Telephone: +45 9635 8080 Telefax: +45 9814 8243